A Review on CFD Analysis of Sail Assisted Ocean Going Ships

Prasanth K[1]., Dr. M. N. Senthil Prakash[2], Dr. K Sivaprasad[3]
[1]Research Scholar,Department of Ship Technology
Cochin University of Science and Technology,Kochi, Kerala, India
Email-prasanthkrishna27@gmail.com
[2]Professor,Cochin University college of Engineering Kuttanad
Cochin University of Science And Technology, Kerala, India
Email-senthil@cusat.ac.in
[3]Professor,Department of Ship Technology
Cochin University of Science and Technology, Kochi, Kerala, India
Email-sivaprasad@cusat.ac.in

Abstract- Oil prices continue to fluctuate from time to time and the rate of increase of oil price has greatly exceeded the rate of general inflation. The price of marine fuels has multiplied more than 15-fold during the last 10 years and has become the largest component of operating cost of maritime shipping. Even in the face of these fuel cost increases, international ocean born trade gives every indication of continued expansion. Ship owners, naval architects and governments of nations around the world have been moving rapidly to cut marine fuel cost through conservation or the use of alternate energy sources. Thus we now have a great opportunity to explore and utilize renewable energy sources like solar energy,wind energy, wave energy etc. Wind energy is the best option in shipping. It is clearly evident that in order to bring down the operating costs in maritime traffic, expenditure on fuel has to be reduced. For this the only possible solution is to find cheaper and efficient alternative energy sources. Among the accessible energy sources in international waters, wind energy is satisfactory for serving this purpose due to its availability and its easiness in transforming it into other forms.Since it is sustainable, easily accessible, low cost and emission-free, many researches have been initiated in designing wind-sails for the propulsion of marine ships. But in most of the studies, not much attention has been given to the combined effect of ship resistance and sail power on marine propulsion.This article intends to review studies in the analysis of economic aspects of sail assisted ships,investigations for minimization of hull resistance including application of numerical methods in ship resistance calculation,research on use of sail for ship propulsion and studies on sail-sail interaction and sail hull interaction.

Keywords: computational fluid dynamics, sail design, sail-assisted ship.

I. INTRODUCTION

Oil is a vital energy source empowering the international economy and delivering about 95% of the entire energy for fueling global transportation. Like other modes of transport, maritime transport relies heavily on oil for propulsion.The fossil fuels reserves are finite and expensive..The history of shipping starts with the origin of trade and commerce.To day about 90 percentage of the world trade is relied on marine industry. A large proportion of international commodity transportis accomplished through sea. The question is how much rise in oil price rise can the shipping industry afford.Although the emissions produced by the commercial ships are very high, it remains as one of the most unavoidable fossil fuel consumers. Shipping contributes quite a large portion to the total greenhouse gas emissions and other toxic emissions.So it is high time to think about alternate sources of energy for commercial shipping, as the world is facing bigger threats regarding environmental impacts like global warming, green house effect, ozone layer depletion, the depletion of marine life and its effects on human life.

Investigating the possibility of using alternate fuel resources for maritime traffic,it has been found that commercial shipping by wind had practically vanished from the sea during the twentieth century. Wind as a means of commercial propulsion did make a brief and limited come back during the 1980's. A lot of researches have taken place regarding the prediction of performance and propulsive ability of sailing crafts. However, controllability of sailing crafts does not seem to have received much attention in spite of many uncertainties and risks involved in harnessing wind power.It is necessary to predict the course-keeping ,maneuvering characterizes of sailing crafts right from the design stage in order to reduce the risk of collision, grounding etc. Also, in the wake of the recent causalities involving sailing crafts, vessels response to wind gusts is another area of concern which needs to be studied.

The projected cost effectiveness of procuring and operating two sizes of multipurpose sail assisted vessel for commercial fishing in Hawaiian waters was investigated by *karl Samples et. al (1985)* [60]. Investment in two comparable sized diesel-powered vessels was also analyzed for comparison.He found that investment-performance of sail assisted fishing boats is inferior to conventional diesel-powered boat given the fuel price, cost of borrowed capital and vessel acquisition costs. Studies have been done by *Satchwell et.al(1985)*[62] using computer calculations to find out optimum engine capacity and fuel consumptions on ships with or without sail assists.It has been seen that the result can be processed to help ship owners in taking investment decisions in this area. Designers can thus compare various systems in this area and get an idea of acceptable costs. The analysis can also help in looking for interesting markets for sail assisted system**.** The facts mentioned in the publication establishes that wind energyis the best alternate source due to its availability, transformation,sustainability, easy accessibility, low cost and emission-free property**.**Scope of research is on the feasibility of effective extraction of wind energy for use in marine transport**.** The state of art research in the field was investigated by reviewing the publications available in the field.The findings of the review are classified and are described here below.

II. Analysis Of Economic Aspects Of Sail Assisted Ships

In the study *Frank Crane A et. al (1985)*[59] analyzed the cost of motor vessels verses sail assisted vessels comparing the costs to operate and maintain their propulsion systems. His study shows that the economy of sail assisted fishing vessel depends on both environmental conditions and types of operations carried out. He also observed that energy efficiency does not necessarily equate to the economic efficiency at all times but it depends upon the types of operations performed by the ship.

In the study of *Chung et. al (2005)* [45],a two-objective model was constructed to find the sailing frequency and optimal ship size.They also analysed the shipping details of very large ships by curtailing shipping cost and inventory costs. The results indicate the Pareto optimum solutions of the two-objective models. And also optimized ship size and sailings frequency with respect to each level of inventory cost and shipping cost. The analysis indicates that optimal ship size is favours at large route flows. Moreover the economies and practicability of using extremely large ships tend to increaseas the port efficiency enhances.when shipping distance increases the ports of call decreases which comparatively decreses the costs of large ships.Review of the above publications shows that it has become inevitable to reduce the usage of fossil fuels. By utilizing renewable energy such as wind, wave, solar, etc can reduce the dependency of the fosil fuel.so many reserch effort has been done on the propulsion of ships by utilizing wind. Advances in this fields are investigated and are presented.

III. Research Publications On Investigation And Minimization Of Hull Resistance

Another method of reducing the consumption of oil is to make the hull resistance minimum. The evaluation of resistance of ships is an important challenge in designing a ship hull. Evaluation of the resistance of water to a moving ship is the most versatile issue because of its lucidity in formulation and difficulty of getting a solution. This area has been studied by many researchers for the last one century.The true history of this particular issue is honored with great names and stupendous ideas it has been described in the book *Tupper E C* [55] that, total resistance of water to the movement of ship is the sum of the hydrodynamic forces affecting the wetted surface of a ship's hull. A part of this force depends on the viscosity of the fluid, effects of friction and the normal stress caused due to viscosity.The other part depends on gravity and gives a wave component of resistance around the ship's hull. In the study of *Lewis U et. al (1989)*[58] a two wave system is considered using semi-infinite wedges. First wave is produced by the wedge located at the bow and the second wave by the wedge located at the stern. The two wave system moves along with the ship hull.Experimental results taken by is set to be more reliable and acceptable . The conventional method that was used for determination of hull resistance is the towing tank test.

Shahjadu bin T et. al (2006)[43] in their paper,mentioned that the practical application of the Computational Fluid Dynamics (CFD) for anticipating the flow pattern around ship's hull had made great progress in the last few years. Today CFD tools play an important role in the designing of ship's hull. CFD has been used for the analysis of ship resistance, sea-keeping and maneuvering parameters for various hull forms. This is a very significant tool used in the primary and final stage of design of the hull. Resistance analysis based on CFD simulation has now become a pivotal factor in the evolution of new economically efficient and environmental friendly ship hull forms.

3.1. Numerical methods in resistance calculation (Computational fluid dynamics)

The study by *Scott Percival A et. al (2001)*[52] using CFD as a tool was to compare the calm-water drag of a series of hull forms and to define an 'optimized' mono-hull ship for which the total calm-water drag is minimum. The friction drag is calculated using the classical ITTC formula. Comparison between theoretical predictions and experimental measurements for a series of eight hull forms were done. The results show that despite the extreme simplicity of the method used to estimate the friction drag and the wave drag, it was able to rank the drags of a series of hull forms roughly in accordance with experimental measurements. *Karl K et. al (2005)*[49] used CFD for the numerical propulsion simulation. RANSE solver for the viscous flow computation, an EULER solver with and without propeller for the determination of propeller induced velocity and potential quasi continuous method for propeller calculations were used. The effective inflow to the propeller is obtained by subtracting the induced velocity from total velocity, obtained by RANSE computation under the propeller influence. Comparison of the result with experimental measurements shows satisfactory agreement.

Propeller open water test, resistance test, and propeller ship hull interaction prediction for a ship's resistance and propulsion performance was done using CFD techniques by *Bhanuprakash et. al (2010)*[24]. Unstructured meshes and structured meshes were used in the propeller region of a ship and remaining part of the hull and its domain respectively.K-epsilon and K-omega turbulence viscous models were employed for simulation. The computational results were validated by comparing it with the existing experimental data.Also Drag analysis was done by Aktar *et. al(2010)* [21] using CFD simulation, on two conventional hull models namely Wigely and Series 60. In this three-dimensional Finite Volume Method has been applied to determine the drag coefficient. The numerical solutions of the governing equations were done using commercial CFD software package FLUENT to compute drag coefficient at different Froude numbers. Velocity vectors as well as contours of pressure distribution on the hull have also been displayed graphically. In his study,computed results show good concurrence with the experimental measurements.In the work of *Choi et. al (2010)*[25] computational tools has been used to scrutinize the speed performance of different types of commercial ships including resistance and propulsion characteristics. For this, eight commercial ships built in the last decade are selected. The limiting streamlines on the hull, wave characteristics around the model ship, and the wake characteristics on the propeller plane were also investigated. After completing the computations, a series of model tests were conducted to evaluate the accuracy of the computational predictions. The predictions clearly disclose the differences in the resistance and propulsion characteristics of different types of commercial ships.Wave characteristic around the model ship and on the propeller plane was also investigated.A series of model tests were done in order to evaluate the accuracy of the computational prediction.It highlights the difference between resistance and propulsion characteristics of different type of commercial ships.

Unsteady Reynolds average Navier-Stokes equation was used by *Shi et. al (2012)* [16] in the calculation of sinkage and trim effect of a steady advancing surface ship, employing volume of fluid method for the treatment of free surface. The sinkage and trim is predicted using dynamic mesh technology and motion of ship is controlled using 6 degree of freedom at seven Froud numbers.The predicted result and experimental result data were compared which showed good agreement. For the resistance of ship model, various numerical solutions were carried out. On the basis of these simulations, the numerical solution and those factors that affect the resistance of the ship were analyzed.*Aktar et. al (2013)*[10] in their study analyzed a conventional model named Wigely parabolic hull in steady-state condition. In the study, the numerical solutions of the governing equations have been obtained using commercial CFD software package FLUENT. Model tests conducted with these two models were simulated to measure various types of resistance coefficients at different Froude numbers. The similarities in the numerical and experimental result depicts that the implemented code represents the free surface elevation around the Wigely parabolic hull precisely. The computed values and experimental measurements were found to be similar.

The resistace induced on the ship during its motion was calculated and validated for several ship models by *Seo M et. al (2014)*[4].They used Rankine panel method and F V based Cartesian method for their analysis. Result obtained using asymptotic formulas were also compared with the obtained result.They Concluded that grid converge test using Rankine panel method required more panels in short wave region. When compared to the panel used for long wave region,due to the spatial variations of physical variables such as wave elevation,the panel used for short wave region were less severe on the body. More panels had to fixed on the bow and stern. All computational evaluation pointed out the increase in resistance at the short-waveregion when the vessel has a blunt body. *Senthil Prakash M.N and Binod Chandra (2013)*[7] has estimated the Shallow Water Resistance of a River-Sea Ship Numerically using CFD. *Hareesh Y. S., Prasanth K. and Senthil Prakash M. N* (2017)[2], in an effort to reduce the hull resistance,tried bubble injection and has done a CFD analysis of drag force reduction in ships by micro bubble injection. Also *Hareesh Y S & M N Senthil Prakash (2020)*[1] has conducted reviews on skin friction drag reduction by micro bubble injection.

Senthil Prakash.M.N and Deepthi.R.Nath (2012)[14] has applied computational method for determination of open water performance of a marine propeller. Even if by virtue of the complicated geometry of the rotating propeller,the simulation is difficult, the open water performance evaluation was a success. In determination of openwater characteristics, only propeller is considered and the propeller will receive un disturbed water flowing in to it. *M .N Senthil Prakash and V Anantha Subramanian 2010* [27] has done the Simulation of propeller- hull interaction using the RANSE solver FLUENT. In it the propeller was treated as a disk, giving the physical effect of the propeller. Simulation of self propulsion of the ship hull propeller combination is very difficult, considering the complicated shape and rotation of the propeller which is working at the aft of a large hull placed in a very large domain. But by following this method of representation of propeller in the simulation, the simulation could be simplified and the requirement of computing power could be reduced. *G. Dhinesh, K. Murali and V. Anantha Subramanian (2010)*[28] has done the estimation of hull-propeller interaction of a self-propelling model hull using a RANSE solver, through a different mesh technology used in the commercial CFD software pack, Star CCM plus.*Senthil Prakash.M.N and V.Anantha Subramanian 2008*[39] has simulated propeller-ship hull interaction using an integrated VLM/RANSE solver. *Senthil Prakash.M.N, V Anantha Subramanian 2009*[35] has simulated the propeller hull interaction by a body force based simulation procedure. *M. N.* Senthil Prakash *and Subramanian V.A 2010*[26] has derived an optimized propeller by coupling *VLM* and RANSE Solver Method. A B-Series propeller was chosen for propelling a candidate hull (KCS hull) and the geometry of the B series propeller the KCS ship hull was modified and optimized.

3.2 Research effort on Investigation towards use of sail for ship propulsion

Two current Stanford Yacht, related to the analysis and design of upwind sails in *Shankaran A et. al (2000)*[53] also develop efficient and robust sail and hull shape optimization methods. They concluded that, the computed lift and induced drag for a main sail with elliptic plan obtained from NASTRAN match with results from lifting surface theories. The flow solver is more efficient than commercial solvers and highly suitable for the computationally intense viscous sail flow calculations.*Viola I M, et. al (2008)*[40] conducted feasibility study for trustable and affordable CFD analysis of aerodynamic indices of racing sailing yachts. Concluded the feasibility of very large parallel CFD processing,that confirm the usefulness of computational approaches as trustable and affordable tools for design .Hypothesis testing gave more and more complementary to the necessary experimental analysis. ADONF software package, to do analysis, design and optimization of complex interacting sails configurations with viscous CFD through automated RANS simulations were studied by *Chapin V G, et. al (2008)*[36] they Concluded that ADONF is a robust, high performance, optimized fluid flows, in the field of the aerodynamic of sailing boats.And also formulates sail shape and trimming optimization in upwind and downwind sailing conditions.

Collie S J,et. al (2002)[51] conducted a Computational Fluid Dynamics based parametric design study of two-dimensional America's Cup Class (ACC) downwind sail section. Finally concluded that CFD can be utilized to obtain better understanding in flow topology and different sail designs and not just for performance prediction. *Alza P A, et. al (2010)* [22] developed a methodology for studying racing yacht sails in upwind conditions by combining full scale measurements with 3D RANS simulations. They concluded that A CFD code is a cost-effective tool for the performance prediction of a sailing yacht and also can be used with remarkable accuracy.A methodology developed by *Wilson PA et. al (2010)* [30] for studying modern square head rig sail in upwind conditions by combining wind-tunnel measurements with 3D RANS simulations concluded that, an extreme accuracy obtained in the acquisition of the flying shapes.And by modeling the hull in addition to the mast, sails improvement producedin term of lift and drag.

A study of Shapes and performance of IMS type sails by *Matsuyama T,et. al (2009)* [33] measured upwind condition using CFD. They concluded that sail shape database and the comparison with the numerical calculations provide a good benchmark for sail performance analysis of the upwind condition for IMS type sails.The paper by *Siulisetyono A, et. al (2010)*[29] described the behaviour of rigid and untwisted sail model in the fluid dynamic analysis. The two-dimensional and three-dimensional wind sail was modelled numerically using CFD technique to determine the influence of the main sail design variables such as draft, camber and angle of attack due to the force coefficients of sail. The 10-sail models were developed, and each sail model was simulated at the variation of the angle of attack such as 15^0, 20^0 and 25^0 respectively. the numerical results were validated by the results of wind tunnel model testing in term of drag and lift coefficient. The simulation showed the design of sail models, which generated maximal efficiency and the most optimal thrust force, were the sail models with camber value 20% and 45% of draft.

Focused the study based on *Riotte M et. al (2014)*[5] using RBF mesh morphing and CFD to explore different trims of yachts sails, the morphing approach is so fast that a real virtual wind-tunnel with a real-time Velocity Prediction Program can be implemented.In *Rojas Let. al (2008)* [37] **c**onducted the study of air flow around a yacht sail and also effect of the mast on the performance of a sail.They Concluded that the influence of the mast is focused on the leeward side of the mainsail where there is a turbulent detachment of the flow and also mast reduces the efficiency of the mainsail.*Banks J et. al (2010*) [23] conducted dynamic wind tunnel tests of an asymmetric spinnaker model passing through a constant wind speed. They concluded that the quasi static heel force data provides a better representation of a real-life gybe and impact of the dynamic force model gives a reduction of boat speed throughout the gybe.*Flay R, et. al (2011)*[19] conducted a full-scale tests, wind-tunnel tests and computational fluid dynamics and concluded that full-scale tests are mostly performed for comparing the performance of two boats. wind tunnel tests used to measure aerodynamic forces and numerical methods are used to compute the pressure distribution on sails.Also Pressures computed numerically were in very good agreement with pressures computed in the wind tunnel.

A study conducted by *FranciscoPerez A et. al (2012)*[17] on sail performance by means of computational analysis and wind tunnel test and also conducted research for saving energy resource. Finally concluded that significant energy can be saved by using the super structure as a sail which leads to a reduction of fossil fuel consumption and also reduction of greenhouse gas emission.In *Bot P et. al (2012)*[15] paper, they explained CFD simulations of three-dimensional sails performed with a RANS approach and validated with local surface pressure measurements obtained with model-scale pressure-tapped rigid sail in an open jet tunel.They concluded that the model, with one free parameter to be determined, give better results. The multiple criteria optimization and numerical computation of a sail fiber reinforced composite model was used to increase the performance of a yatch under wind action. They concluded that by using multiple criteria and optimization tools to improve the quality, efficiency and performance of a mainsail and numerical optimization procedure advanced in nonlinear finite element modeling.

Viola I M et. al (2013)[11] conducted wind tunnel experiments on a 1:15^{th} model-scale sailing yacht modeled with RANS and DDES. They concluded that DDES seems to be able to predict sail performance more accurately than RANS. Most of the aerodynamic load was carried by the spinnaker, which experienced the up wash of the mainsail. Forces and pressures were almost independent from the time and space resolutions in a work.Two mainsails, a conventional TP52 style and an alternative was chosen to be closer to the IRC default girthvalues based on upwind performance in the study done by *Robert et. al (2013)* [9] Result obtained that a TP52 style mainsail is not optimal under IRC. The combination of rating reduction and predicted performance suggest that an alternative mainsail with larger P and E with girth values closer to the IRC default values is a superior choice for an IRC 52.Work by *Yeongmin Jo et. al (2013)* [8] had conducted a high-fidelity two-dimensional CDF analysison Sail model NACA 0012. To improve the aerodynamic performance of the wing-sails, they included a flap system to generate high lift. They computed the best angle of attack for different wind angle. This paper is considered as the base paper for the work.

3.3. Research effort on Investigation towards sail-sail interaction and sail hull interaction

In the study done by *Fujiwara N et. al (2005)* [48] conducted wind tunnel testing to investigate Sail–sail and sail–hull interactions between the four rectangular and triangular type hybrid-sails with fully loaded and ballasted conditions of a bulk carrier by using a graduated arrangement . This experiment was conducted to reduce the CO_2 production of diesel engine exhaust relative to solving the global warming problems. They Concluded that the rectangular soft sail system is generally more efficient than the triangular soft sail system because the interaction effect of the rectangular sails is smaller than that of the triangular sails. And the rectangular sails induce more driving force than the triangular sails.The steady-state effects of a bulk carrier ship with a new type of hybrid-sails were investigated using MMG simulation method and experimentally determined key hydrodynamic and aerodynamic coefficients. The benefits of the sail-assisted bulk carrier ship for an expected reduction of BHP and FOC when operating in North Pacific Ocean were also presented.They concluded that U = 13.5 knots, with rectangular type hybrid-sails gains 11.7% of the required thrust from the sails andtriangular type hybrid-sails, gain is 8.3%. These gains increase with increasing wind speed. BHP results for sails set in the graduated and goosewinged arrangement for U = 13.5 knots, indicate that the expected value of BHP gain from the hybrid-sails is 12.4% for rectangular type sails and 9.2% for triangular sails.

Higher MCR limitation will reduce the fuel consumption benefit to an average level of 13.3% for the preferred rectangular soft sails was found in a study conducted by *Fujiwara N et. al (2005)* [47].*Pravesh Chandra S et.al (2009)* [24] designed a computer programme, which by feeding the data regarding wind velocity, wind direction and ship-motion direction,will provide the best wing-sail position and fuel savingfor commercial ships.They have calculated net fuel saving in certain international shipping routes, for instance, from Mumbai in India to Durban in South Africa.There estimates show that about 8.3% diesel fuel can be saved by utilizing the wind.. It is established that the use of modern wing sail in ship propulsion will reduce diesel fuel consumption considerably. These wing sails can be retrofitted on existing ships. From the above estimation it is clear that the use of wind energy in ship propulsion may save 8.3% of fuel.

Ouchi et. al (2011)[20] conducted CFD simulation on nine wing sail system and total thrust is also estimated in case of upwind, beam and downwind on the wind powered vessel characterized “motor-assisted sailing vessel”.The paper concluded that the deflection and stress calculation for that system is reasonable and economical for a very large merchant vessel such as a 180,000DWT cape-size bulker and also suggested that significant power from such huge sails can decrease fuel oil consumption substantially and more than 50% of propulsion energy can be saved by consuming the wind energy at sea in the Pacific Trade Wind Region.

Several aspects of sail-assisting technology by *Robert et. al (2013)*[9].In this paper,they introduces sail type selection and experimental results of arc sail models. Thrust force coefficient, drifting force coefficient, lifting force coefficient, resistance coefficient and rotating torque coefficient of the sail model are discussed and optimal sail rotated angle is calculated in the paper .From the experimental result a control mechanism and the material of sail structure are designed for the sail operation of a ocean-going bulk carrier.Comments and recommendations are finally discussed for the further application of the sail onboard ship. According to the wind tunnel test result, The maximum thrust coefficient and the corresponding drifting force coefficient of the sail model are between 25 deg and 45 deg.A multi-mast, rotary, stacking-arrangement sail structure was designed for ocean-going ships, which could be useful for improving the stability of sail-assisted ships. Based on the calculation method proposed, a model of sail-assisted ship was made, and its stability was checked. The results verify that the proposed stability criteria of sail-assisted ship are feasible.

IV. CONCLUSION

The review of the literature discussed above describes the research efforts in the fields of economic aspects of sail assisted ships, investigation and minimization of hull resistance, investigation towards use of sail for ship propulsion, Investigation towards sail-sail interaction and sail- hull interaction. Literature discussed in the section is investigation and minimization of hull resistance using computationlmethods.Which is almost similar to the initial part of the objective of current work.CFD analysis of sail alone for various operating conditions were done and reported in *Yeongmin Jo et.al (2013)* [8] . A modified NACA 0012 model with a flap was used flap as geometry. They determined the best angle of attack at different operating condition. The flap is also optimized using a high fidelity two dimensional CFD analysis on Ansys Fluent. This paper is considered as a base paper for this project. In this NACA 0018 is used because it has much performance than NACA 0012. The first step will be to optimize the flap angle at different chord length at standard operating conditions. Then multi sail condition were studied and optimized and the angle of attack will also be optimize to best suit the application at different operating conditions.The stability conditions will be check analytically but design and analysis of sail configuration subject to transverse stability constraints with sail- hull interaction using CFD is more realistic but is not done so far. It is one of the main objectives of the present work..Not much attension had given to the sail hull combination using CFD.The third objective computation of driving force and heeling force generated by the selected hull- sail combination for a range of apparent wind speeds and apparent wind angles using CFD. which is very much important in finding out the power given by the sail maintaining stability of the ship.

.

REFERENCES

[1]. Hareesh Y S & M N Senthil Prakash (2020) Reviews on Skin Friction Drag Reduction by Micro Bubble Injection Volume 14 Issue 1 June 2020

[2]. Hareesh Y. S., Prasanth K. and Senthil Prakash M. N (2017) CFD Analysis of drag Force Reduction in Ship byMicro Bubble Injection, Advance in Technology, Engineering and Computing , SCMS School of engineering and Technology, Ernakulam – A Multinational Colloquium – 2017 June 22-23 pp 237-248.

[3]. Yihuai Hu1andJianhai He (2015), “Sail Structure Design and Stability Calculation for Sail-assisted Ships”. Marine Engineering Frontiers (MEF) Vol 3, 2015.

[4]. Seo M., Kyung Kyu Yang, Dong Min Park,andYonghwanKim(2014) "Numerical analysis of added resistance on ships in short waves". Journal of Ocean Enineering, Vol 87, 97–110.

[5]. Riotte M, Biancolini ME and Viola IM (2014) "sails trim optimization using CFD and RBF mesh morphing, Journal of Computers and Fluids,Vol 93, 46-60.

[6]. Heather Peng,Shaoyu Ni, Wei Qiu(2014) "Wave pattern and resistance prediction for ships of full form". Journal of Ocean Engineering, Vol 87, 162–173.

[7]. Senthil Prakash M.N & Binod Chandra(2013), Numerical Estimation of Shallow Water Resistance of a River-Sea Ship Using CFD, International Journal of Computer Applications, 71(5),33-40,May2013.

[8]. YeongminJo(2013), "Aerodynamic Design Optimization of Wing-Sails" Fluid Dynamics and Co-located Conferences, 31stAIAA Applied Aerodynamics.

[9]. Robert, Dave ArmitageandB.AdolfoCarrau.(2013),"Mainsail Planform Optimization for IRC 52 Using Fluid Structure Interaction",The 21st Chesapeake Sailing Yacht Symposium Annapolis, Maryland,

[10]. Aktar, Goutam Kumar S and Abdul AlimMD(2013) "Numerical Computation of Wave Resistance Around Wigely Hull Using Computational Fluid Dynamics Tools, Advance Shipping and Ocean Engineering,Vol 2, 84-95.

[11]. Viola IM, BartesaghiSandVan-Renterghem M (2013) "Delayed Detached Eddy Simulation of Sailing Yacht Sails". The Third International Conference on Innovation in High Performance Sailing Yachts, Lorient, France.

[12]. Bot M, Viola IMandRiotte (2013) "Upwind sail aerodynamics: A RANS numerical investigation validated with wind tunnel pressure measurement" International Journal of Heat and Fluid Flow,Vol 39, 90-101

[13]. KaturiSamarpana, Ajay KonapalaandDuvvada Ramesh (2013)"Computational investigation of free surface flow around a ship hull". International Journal of Application or Innovation in Engineering & Management (IJAIEM),Vol2, 533-541.

[14]. Senthil Prakash.M.N &Deepthi.R.Nath(2012) A Computational Method for determination of open water Performance of a Marine Propeller, International Journal of Computer Applications, 58(12),1-5,Nov.2012.

[15]. Bot P, ViolaIMandRiotteM (2012) "Upwind sail aerodynamics: A RANS numerical investigation validated with wind tunnel pressure measurement" International Journal of Heat and Fluid Flow, Vol 38, 90-101.

[16]. Shi Ming Wu, Bo Yang, Xiao WangandZuochao Wang. (2012) "Resistance Calculation and Motions Simulation for Free Surface Ship Based on CFD", Procedia Engineering,Vol 31, 68 – 74.

[17]. Francisco Perez A, Patricia, A, Luis P, Rojas B and Alberto Torres B (2012) Procedia - Social and Behavioral Sciences 48 "EU- Cargo Xpress: Wind Propulsion Concept" pp 1314 – 1323, Victoria, 20844, Madrid.

[18]. Ahmed YM (2011). "Numerical simulation for the free surface flow around complex ship hull form at different Froude numbers". Alexandria Engineering Journal, Vol 50, 229–235.

[19]. Flay R, Richard GJ, Viola IM (2011) ´Sail pressures from full-scale", wind-tunnel and numerical investigations, Ocean Engineering,Vol 38, 1733–1743.

[20]. Ouchi, K Kiyoshi U, and Akihiro K (2011) "Huge Hard Wing Sails for the Proposer of Next Generation Sailing Vessel". Second International Symposium on Marine Propulsorssmp'11, Hamburg, Germany.

[21]. Aktar(2010) Drag analysis of different ship models using computational fluid dynamics tools., Proceedings of MARTEC 2010 "The International Conference on Marine Technology".Vol 10, 11-12.

[22]. Alza PA, Pérez Rojas, L Zamora Rodriguez A (2010) "The Effect of the Mast on Sail Performance"IV Symposia International de Diseñoy Production de Yates de Motor y Vela, pp 159-168.

[23]. BanksJ,Webb A, SpenkuchT, TurnockSR(2010)Measurement of dynamic forces experienced by an asymmetric yacht during a gybe, for use within sail simulation software.ProcediaEngineering,Vol2, 2511–2516 Southampton, UK.

[24]. BhanuprakashTV, Duna SridharK, DasHN(2010) Frictional Resistance Calculations on a Ship using CFD, International Journal of Computer Applications,Vol11,8882-8887.

[25]. Choi JE,MinKS,KimJH ,Seo H(2010) "Resistance and propulsion characteristics of various commercial ships based on CFD results", Ocean Engineering,Vol 36, 549-566

[26]. M. N. Senthil Prakash, Subramanian V.A,(2010) Optimization of Propeller by Coupled *VLM* and RANSE Solver Method, Melbourne, Florida, Oct 13-15, 2010 pp 250-260.

[27]. M.N Senthil Prakash and V Anantha Subramanian(2010), Simulation of propeller- hull interaction using RANSE solver, International Journal of Ocean and Climate Systems, 1(3),189-203,2010

[28]. G. Dhinesh, K. Murali & V. Anantha Subramanian (2010) Estimation of hull-propeller interaction of a self-propelling model hull using a RANSE solver, Ships and Offshore Structures, 5:2, 125-139,

[29]. Siulisetyono A, Nasurudeen A (2010) "Wind sail analysis using CFD simulation", Proceedings of MARTEC 2010.

[30]. Wilson PA (2010) "A comparative study between wind-tunnel experiments and RANS simulations of modern square headed main sails". International Journal of Small Craft Technology,Vol 151, 1-10.

[31]. Pravesh Chandra Shukla and Kunal Ghosh (2009)."Revival of the Modern Wing Sails for the Propulsion of Commercial Ships"International Journal of Mathematical, Computational, Physical, Electrical and Computer Engineering Vol: 3,3-6.

[32]. Espinosa JandOrtigosa I (2009) "A Calculation/Simulation System of Sails Communicated IN Real Time With Wireless Sensors" International Conference on Computational Methods in Marine Engineering MARINE.vol 9,46-51.

[33]. Matsuyama T, Tahara Y, Fukasawa TandKatorM (2009) "Sail Performance Analysis of Sailing Yachts by Numerical Calculations and Experiments", Fluid Dynamics, Computational Modeling and Applications,vol 7,67-72.

[34]. Viola IM (2009) "Downwind sail aerodynamics:A CFD investigation with high grid resolution", Ocean Engineering,vol 36, 974-984.Auckland, New Zealand.

[35]. Senthil Prakash.M.N, V Anantha Subramanian (2009) Body force based simulation of propeller hull interaction, Proceedings of the third international conference in ocean Engineering, ICOE-2009,Feb 1-5 2009 pp 532-542

[36]. ChapinVG,NeyhousserR,DulliandG and ChassaingP(2008) "Design Optimization of Interacting Sails Through Viscous CFD" In: International Conference on Innovation in High Performance Sailing Yachts,16,541- 560.

[37]. Rojas and L. Zamora Rodriguez A (2008). "Computational Study of Sail Performance on racing yacht sail".Journalof Wind Engineering and Industrial Aerodynamics, 96, pp 143- 165.

[38]. Collie CJ, Jackson PS, Gerritsen S, Fallow JB (2008)"Two-Dimensional CFD-Based Parametric Analysis of Downwind Sail Designs"High Performance Yacht Design Conference, Auckland NZ.

[39]. Senthil Prakash.M.N & V.Anantha Subramanian (2008) Simulation of propeller-ship hull interaction using an integrated VLM/RANSE solver modeling, Proceedings of Sixth International Conference On High-Performance Marine Vehicles, Naples, Italy, 18-19 September 2008, pp 334-345

[40]. Viola IM, Raffaele Ponzini I and Daniele RocchiN (2008) "Analysis of Aerodynamic Indices for Racing Sailing Yachts: a Computational Study and Benchmark on up to 128 CPU's". The twentieth in the high-level international series of parallel computing in computational fluid dynamics, vol 8,61-69.

[41]. TimmerWA(2008)"Two-dimensional low-Reynolds number wind tunnel results for airfoil NACA 0018".Journal of Wind Engineering,vol32, 525–537.
[42]. Ciortan C, Guedes Soares, M. (2007) "Computational study of sail performance in upwind condition", Ocean Engineering,vol 34, 2198-2206.
[43]. Shahjada bin T, Gazi bin M, KhalilHandSMIkhtiar bin Mahmud B(2006) "Computation of wave-making resistance of wigley hull form using michell's integral" Dhaka-1000, Bangladesh.
[44]. Yoo J, Kim TH (2006). "Computational and experimental study on Performance of sails of a yacht", Ocean Engineering vol 33, 1322–1342.
[45]. Chung Ing H(2005). "Shipping Economic Analysis for ultra large container". Journal of eastern Asian society for Transportation studies, Vol 6, pp 936-951.
[46]. Clauss GF, Heisen W (2005) CFD "Analysis on the flying shape ofmodern yacht sails", Maritime Transportation and Exploitation of Ocean and Coastal Resources, vol 14, 87–94.
[47]. FujiwaraN, Grant E,HearnN, Fumitoshi K, MachinUenoR, and Yoshimasa MinamiN (2005), "Fast sail assisted feeder container ship", Journal of ocean engineering,vol15;425-430.
[48]. FujiwaraN, Grant E HearnN, Fumitoshi K, Michio UenoR, and Yoshimasa Minami,N(2005) "Sail–sail and sail–hull interaction effects of hybrid-sail assisted bulk carrier". Journal of Marine Science and Technology, volume10, pp 82–95.
[49]. Karl K and chao Y (2005) "Numeric propulsion simulation for the KCS Container ship", Journal of Applied Ocean Research ,Vol.6 pp 483-488.
[50]. Toshifumi Fujiwara,N, Grant, E HearnN, Fumitoshi K, Michio UenoR and Yoshimasa MinamiN (2005) "Steady sailing performance of a hybrid-sail assisted bulk carrier". Journal of Marine Science and Technology, vol 10, 131–146.
[51]. Collie SJ, Jackson P, Gerritsen M (2002) Validation of CFD Methods for Downwind Sail Design. In proceedings of the High-Performance Yacht Design Conference, Auckland, New Zealand.
[52]. Scott Percival, A(2001)., Hydrodynamic optimization of ship hull forms. Applied Ocean Research,vol 23,337-355.
[53]. ShankaranA, Tyler Doyle I, MargotG, GianlucaIandAnthonyJ(2000) "Improving the Design of Sails using CFD and Optimization Algorithms"High Performance Yacht Design Conference, Auckland NZ,vol 5, 4-6.
[54]. Miyata H and Lee YW(1999). "Application of CFD simulation to the design of sails". Journal of Marine Science and Technology,vol 4, 163–172.
[55]. Tupper EC(1998) "Introduction to Naval architecture", Butter worth. Heinemann.
[56]. John v Anderson (1995) "Computation of Fluid Dynamics".
[57]. VersteegHK, MalalasekaraM(1995). "An introduction to computational fluid dynamics".
[58]. Lewis U(1989) "Principles of Naval architecture", vol 3, SNAME, New York.
[59]. Frank CraneA, David A,LauderdaleH (1985) "Aquarian Research Inc An Analysis of Sail – Assist in Today's Fisheries", Fort Lauderdale Florida.
[60]. Karl SamplesC (1985) "An economic appraisal of sail- assisted commercial fishing vessels in Hawaiian waters", Marine fisheries review.
[61]. SatchwellCJ(1985),"Wind ship Technology", proceeding of the international symposium on wind ship technology (WIND TECH'85), Southamton,U.K,Elsevier,Amsterdam-oxford-New York,Vol-4B.
[62]. SatchwellCJ(1985),"Wind ship Technology",proceeding of the international symposium on wind ship technology (WINDTECH'85), Southamton,U.K,Elsevier,Amsterdam-oxford-New York,Vol-4A.
[63]. HarvardSA(1983) "Resistance and propulsion of ship", John Wiley& son.